EM433 Computer-Aided Manufacturing
Lab 3 - Numerical Control Programming

Objective

One component of NC machining is the part program that defines the part geometry and machining parameters and sequence to the machine controller. Most controllers use some variation of *G-code* as the part programming language. Like any programming language, there are certain rules and conventions that must followed in order for the program to run properly. This laboratory exercise is designed to make you familiar with G-code by writing a simple program. It is not the intent of this lab to make you a skilled part programmer, but it is important that you understand how a part program works so that you can debug the programs that I-DEAS generates for you.

Part Programming

A part program is made up of a sequence of program statements called blocks. Each block is composed of one or more "words". The words can indicate a machine action such as moving the cutter to a specified location, an auxiliary function such as turning on the spindle, or it may be a data word that indicates a coordinate or a speed. Most G-words are common amongst controllers, but there are exceptions so it is important to consult the operating manual for the particular controller being used.

A typical block from a program for the Heidenhain controller on the Wells-Index mill in TSD is listed below.

N100 G01 X+4.5000 Y+2.5000 Z-.1250 S1000 F100 M03

The first word is usually an N-word, which is simply a sequence number. Usually the sequence numbers are incremented in steps of 5 or 10 to allow additional lines to be inserted if necessary. Next is a G -word or a preparatory function code. These commands control the motion of the table slides and tell the controller where to position itself. In the example shown above, the G01 code is for linear interpolation from the current position to the coordinates indicated by the X, Y and Z words that follow on the line. This motion is conducted at the current programmed feed rate. In the block above, the x, y and z coordinates would be (4.5, 2.5, -0.125). Following the X, Y and Z words are modifiers to set or change the spindle speed and feedrate. These are S and F words, respectively. Finally, a block can contain an auxiliary word or an M-word. These auxiliary words control other machine functions like starting the spindle clockwise (M03), turning the coolant on or off, requesting a tool change or signifying the end of the program. A T-word is necessary to specify the tool number if a tool change is requested.

Many of the G-cods are modal. A modal command is one that stays in effect until it is superceded by another command. The following example lines of code are used to illustrate this principle.

N002 G00 X+1 Y-2 Z+3
N004 Z+.5

N006 G01 Z-.1
N008 X+2
N010 X+2.5 Y-1.5

The first line directs the controller to rapidly position the machine spindle at (1,-2,3) from the current location. The G00 command, for rapid motion is a modal command and will remain in effect until it is cancelled by another G word. Line N004 doesn't contain any G words, only a z-coordinate word. Since rapid motion has been asserted by the G00 from the previous line, the spindle will rapidly advance from the current position (1, -2,3) to (1, -2, 0.5). The next line contains a new G word, G01, which is a command to move to the indicated position, using linear interpolation, at the programmed federate. Therefore, the spindle would be positioned at (1, -2, -0.1). Next it will advance at the programmed feedrate to (2, -2, -0.1) and then to (2.5, -1.5, -0.1).

A list of common Preparatory words and auxiliary function words for the Heidenhain controller are listed at the end of the assignment sheet (or were distributed in class).

Lab Assignment:
Your assignment is to develop an NC part program that will write your name on a 2 in. x 6 in. x 0.25 in piece of aluminum stock. After you have completed the program you will save it on a 3.5 in floppy disk and bring it to TSD to be machined.

1. Layout your name plate on a sheet of graph paper. Determine the coordinates of the start and end point of each line segment as well as the coordinates defining the center of each circular arc.
2. All cuts shall be made at a depth of 0.075 in. or less. The cutter will be a ¼ in. dia. ball end mill so the cut will run 0.115 in. to either side of the programmed path. (A narrower cut is possible by reducing the depth of cut.) For this part program, assume that the origin of the coordinate system is positioned at the lower left-hand corner on the top surface of the stock. The stock will be held in a vise so there will not be any clamps to avoid in the work area. The coordinate system for a mill is a right-hand system, positive x to the right, positive y, away from you, and positive z, normal to the table, going up.
3. The first three blocks should be as follows (don't enter the comments in italics):

N0100 M06	*Tool Change*
N0102 T1 S1000	*Select tool #1, set speed*
N0104 G00 X0 Y0 Z0.25 F100 M3	*Go to (0,0,0), set feed, start spindle*

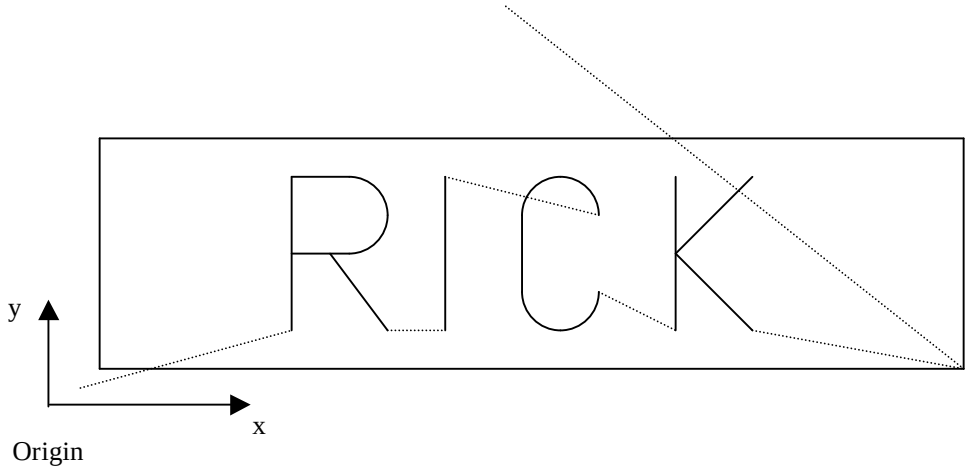
This will position the tool 0.25 in. above the stock at the origin. The speed is set at 1000 rpm, the feed at 10 in/min, with the spindle running. Your next move should be a rapid traverse (G00) to the (x,y) position above your first point. This should be followed by a linear interpolation (G01) move into the stock. A sample program is given below to engrave "RICK" on a piece of stock. Don't forget to retract the cutter straight up before moving to the next letter starting position.

4. Check you program by manually working through all of the programmed moves and plotting the path. Use a red pen for moves above the surface and a dark pen for moves that occur with the cutter engaged in the part.
5. Swap program files with a classmate and verify that the program properly specifies the moves necessary to produce a nameplate. You should trace thru and plot the program in order to verify the coding.
6. Once you are satisfied that the program is correct, submit the program and the verification plot to your instructor for approval.
7. Use Notepad or some other text editor to record the program as an ASCII text file on a 3-1/2 in. floppy disk and submit it to Mr. Jerry Myers in TSD to be machined. You should arrange a mutually convenient time for the machining of your nameplate.
8. When you have completed your project, show it to me so that I may record a grade for this assignment.

SAMPLE PROGRAM

N0100 M06	*Tool change*
N0102 T1 S1000	*Use Tool #1*
N0104 G00 X0 Y0 Z0.25 F100 M3	*Rapid to origin, 0.25 above stock*
N0110 X1.6 Y0.5	*Rapid to start of letter "R"*
N0115 G01 Z-0.0750	*Plunge into part at feed rate*
N0120 Y1.5000	*Vertical stroke of "R"*
N0125 X1.9500	*Feed across top of "R"*
N0130 X1.9500 Y1.0000 G02 I1.9500 J1.2500	*Feed arc cw*
N0135 G01 X1.6000	*Feed across middle*
N0140 X1.9500	*Retrace*
N0145 X2.2000 Y0.5000	*Feed leg of "R"*
N0150 G00 Z0.2500	*Rapid clear of stock*
N0155 X2.6000	*Rapid to bottom of "I"*
N0160 G01 Z-0.0750	*Feed into stock*
N0165 Y1.5000	*Feed up "I"*
N0170 G00 Z0.2500	*Rapid clear of stock*
N0175 X3.5000 Y1.2500	*Rapid to start of "C"*
N0180 G01 Z-0.0750	*Feed into stock*
N0185 X3.0000 Y1.2500 G03 I3.2500 J1.2500	*Top arc of "C" ccw*
N0190 G01 Y0.7500	*Feed to start of bottom arc*
N0195 X3.5000 Y0.7500 G03 I3.2500 J0.7500	*Bottom arc of "C" ccw*
N0200 G00 Z0.2500	*Rapid out of stock*
N0205 X3.9000 Y0.5000	*Rapid to start of "K"*
N0210 G01 Z-0.0750	*Feed into stock*
N0215 Y1.5000	*Feed up stroke of "K"*
N0220 Y1.0000	*Retrace halfway down*
N0225 X4.4000 Y1.5000	*Feed upper slant line*
N0230 X3.9000 Y1.0000	*Retrace to beginning of slant line*
N0235 X4.4000 Y0.5000	*Feed lower slant line*
N0240 G00 Z0.2500	*Rapid out of stock*

N0245 X5.7500 Y0.2500	*Rapid to start of outline*
N0250 G01 Z-0.0750	*Feed into stock*
N0255 X0.2500	*Feed across bottom edge*
N0260 Y1.7500	*Feed up left side*
N0265 X5.7500	*Feed across top*
N0270 Y0.2500	*Feed down left side*
N0275 G00 Z0.2500	*Rapid out of stock*
N0280 X3.0000 Y6.0000	*Rapid clear of part*
N0285 M30	*Program Stop*



List of Common G-Codes for NC Machining

G00 X Y Z	Rapid traverse to (x,y,z)
G01 X Y Z	Linear Interpolation to (x,y,z) at programmed feedrate
G02 X Y Z I J K R	Circular interpolation to (x,y,z) clockwise, radius R, center located relative to current position by (i,j,k) On some controllers the center may be defined in absolute coordinates (depends on whether G90 or G91 has been invoked). If I,J,K are specified, then R is not. If R is specified, then I,J,K are not.
G03 X Y Z I J K R	Circular interpolation to (x,y,z) counterclockwise, radius R, center located relative to current position by (i,j,k)
G04 F	Dwell for f seconds
G40	Cancel cutter compensation
G41	Cutter compensation on left
G42	Cutter compensation on right
G70	Dimensions in inches
G71	Dimensions in millimeters
G81-89	Canned cycles for drilling, tapping and boring, etc.
G81 X Y Z R F	Rapid to (x,y,r) Drill to depth, z, at feedrate F, rapid out
G90	Absolute coordinates
G91	Incremental coordinates
G94	Feedrate (in/min)
G95	Feedrate (in/rev)
G96	Constant surface speed (ft/min)
G97	Constant spindle speed (rpm)

Most of the G codes are “modal”. This means they are in effect until canceled or changed by another G-code from the same group.

Miscellaneous Codes (M-Codes)

M00	Program stop
M01	Optional stop
M03	Spindle on (cw)
M04	Spindle on (ccw)
M05	Stop spindle
M06 T	Change tool
M07	Coolant on (flood)
M08	Coolant on (mist)
M09	Coolant off
M13	Spindle on (cw), coolant on
M14	Spindle on (ccw), coolant on
M30	End of program